**US Army Corps
of Engineers**
Waterways Experiment
Station

Guide to XMESH

by   John F. Peters, Ronald E. Wahl, Ronald B. Meade, WES
Raju Kala, Mevatec Corporation

19970609 059   DTIC QUALITY INSPECTED 3

Guide to XMESH

by   John F. Peters, Ronald E. Wahl, Ronald B. Meade

U.S. Army Corps of Engineers
Waterways Experiment Station
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

Raju Kala

Mevatec Corporation
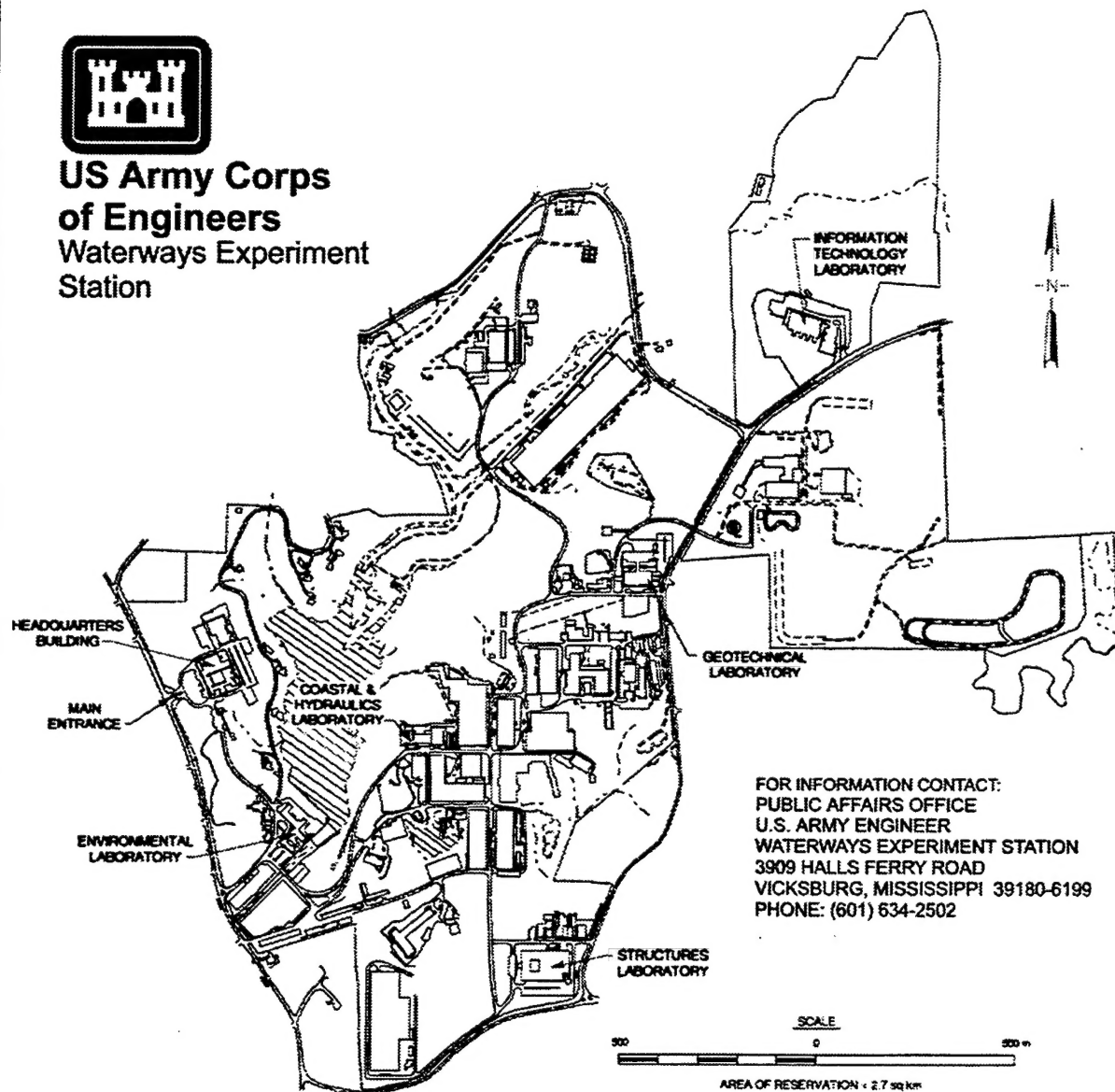3046 Indiana Avenue
Suite 172
Vicksburg, MS   39180

Final report

|DTIC QUALITY INSPECTED 8

US Army Corps of Engineers
Waterways Experiment Station

FOR INFORMATION CONTACT:
PUBLIC AFFAIRS OFFICE
U.S. ARMY ENGINEER
WATERWAYS EXPERIMENT STATION
3909 HALLS FERRY ROAD
VICKSBURG, MISSISSIPPI 39180-6199
PHONE: (601) 634-2502

AREA OF RESERVATION < 2.7 sq km

Contents

List of Figures

List of Tables

Preface

The United States Army Engineer Waterways Experiment Station (WES), under the sponsorship of the Civil Works and Repair, Evaluation, Maintenance, and Rehabilitation (REMR) Research Programs, developed a pre- and postprocessor (XMESH) for use with a finite element program (STUBBS).

The creation of the computer program was primarily the concept of Dr. John F. Peters, Geotechnical Laboratory (GL). Mr. Ronald E. Wahl, GL, was also a principal in the development of the program, and Dr. Ronald Meade, GS-GC, was responsible for the preparation of this users manual. Mr. Raju Kala, Mevatec Corporation, was the primary engineer for the development of graphics features. Messrs. Dallas Barlow and Daniel Olson, contract students, provided extensive computer programming expertise in the initial development of the code.

The project was under the direct supervision of Mr. Robert D. Bennett, Chief, Soil Research Facility, and under the general supervision of Dr. Don C. Banks, Chief, Soil and Rock Mechanics Division, and Dr. William F. Marcuson III, Director, GL.

At the time of publication of this report, the Director of WES was Dr. Robert W. Whalin. The Commander was COL Bruce K. Howard, EN.

1   Introduction

Description

XMESH is a finite element preprocessor, a postprocessor, and a menu-driven input file builder. The program was developed at WES as a productivity tool to be used with STUBBS, a finite element program. XMESH runs on any UNIX based system and is used at WES on the Hewlett-Packard, Silicon Graphics, and the Cray platforms. Additionally, XWINDOWS and MOTIF are software packages which must be present on the system for XMESH to work properly. Although XMESH was written to be used with STUBBS, the XMESH program can be used in a general fashion as a preprocessor to create nodes and elements, and as a postprocessor to provide a visual display of data.

Characteristics

The program is menu-driven. The user enters information using the keyboard and a three-button mouse. The mouse buttons are indicated in the on-screen instructions as MB1, the left mouse button, MB2, the center mouse button, and MB3, the right mouse button.

The general appearance of the program is shown in Figure 1 below. The drawing/display space is surrounded by a rectangular border. Along the top of the border is a line of main menu items oriented horizontally. Along the bottom of the border is a text line, partitioned into three areas. Along the left-most portion of the bottom border area is a command line, into which the user enters information using the keyboard. The right-center portion of the line is used for messages and prompts that provide instructions for the user. The right-most portion of the line is used to display the current mesh coordinates of the mouse pointer.

XMESH has many CAD-like features: zoom and pan, entering data using either the mouse or keyboard, selecting nodes and elements by fencing. In addition, the program has customary windows features. XMESH runs in a window that can be re-sized, minimized to an icon, and restored to a full window by clicking on the icon. Multiple copies of XMESH can run simultaneously, each in its own window. Other windows features include dialog

```
+----------------------------------------------------------------------+
|                              xmesh                                   |
+----------------------------------------------------------------------+
| File  View  Edit  Make  Set  Show  Size  Display  Legends  Batch  Step  Build |
+----------------------------------------------------------------------+
|                                                                      |
|   (drawing area)                                                     |
|                                                                      |
|                                                                      |
|                                                                      |
|                                                                      |
+----------------------------------------------------------------------+
| Command:                          |  (msg area)                |     |
| (coord)                                                              |
+----------------------------------------------------------------------+
```

Figure 1.   Screen area in XMESH

boxes, buttons, and pull-down menus. It is assumed that the users are
familiar with each of these windows features.

When data entry is described, the use of the "Enter" key (←) is indicated
in the text as < CR >. In general, when the user is prompted to enter data
from keyboard, the data entry should be terminated with < CR >.

2   Preprocessing

The creation of a mesh of finite elements can be a daunting task. Most finite element codes either include a preprocessor or presuppose the use of a generic preprocessor to create a suitable collection of elements known as a mesh. Among the usual tasks of a preprocessor are the ability to create nodes and elements, to refine the mesh, and enter boundary conditions. XMESH provides these capabilities and many more features.

Data Input

When the mouse is placed in the border areas, on the top menu, or on the bottom command line, the mouse pointer is an arrow. When the mouse pointer is placed in the drawing portion of the screen, the pointer is a cross. Menu items are selected by the mouse using MB1 and usually terminated by using MB3. However, if necessary, selection of the escape key (ESC) on the keyboard can be used to break out of most menu selections.

The **File**, **Set**, **Show**, and **Edit** menu items will be used in nearly every session. The **File** item creates, loads, and saves files. The **Set** item specifies options that will control, in part, how the other menu items work. The **Show** item is used to turn on/off visual indicators that permit the user to see mesh specifics, such as node numbers, element numbers, boundary conditions, status of the blocking parameter used in mesh refinement, and other useful information. Each of the main menu items will be described in the following sections.

The program is CAD-like and permits the user to select elements and nodes using the mouse. The technique used is "the fence." The user is prompted to set one corner of a region to be fenced. After the first corner is fixed, the mouse pointer (a cross) shows a rectangular area enclosed within a thin black line. The border line is anchored to the first corner and stretches like an rubber band in real time as the mouse is moved. The first corner and the mouse pointer form the diagonal of the rectangle. When the user is satisfied that the fence encloses the proper nodes or elements, the user again selects MB1 and the fence is used to identify those nodes or elements to be affected by some operation. If the user is not satisfied with the first corner, he may press MB2 to start over. Pressing MB3 completes the fencing

process. In the remainder of this document when some XMESH menu selection prompts the user to make a fence, the operation will not be further described other than to say that the user is prompted to make a fence.

Initially, only **File** is active. Once a file is created by making one or more elements or by loading a preexisting file into XMESH, all of the remaining menu items become active with the exception of **Step**.

File Menu

Only a few of the options under **File** are active when XMESH starts. The menu items under **File** appear as:

New
Load Mesh
Load data
Directory
Save
Dump
Animate Steps
Capture Steps
Exit

The options in **bold** are active when the program starts. The active items also appear highlighted (bold) on the XMESH screen.

When **New** is selected, the user is prompted to enter a mesh file name on the command portion of the screen (bottom left). When a file name is entered, the user is prompted, "Mesh coordinates, lower left - X,Y:" Assume that the coordinates 0, 0 are entered. Then, the user is prompted for the "Mesh coordinates, upper right - X, Y:" The message line reads, "Enter the extreme coordinates of the mesh." Assume the user enters 100, 100. If the file name corresponds to an existing file in the active directory, the user is prompted, "Overwrite?" to which the user should respond with (y)es or (n)o. If the user responds no, then the prompt asks once again for the file name. If the user responds yes, then the old file is overwritten with the new mesh data.

Then, the user is prompted for material type number and the message area reads "Material Properties for Solid elements." Assume that the user enters the number 1.

Then, the user is prompted for a "material type title: ." Assume that the user enters "clay."

Then, the user is prompted for the approximate center of the element. The user selects the position either by using the mouse or by typing the coordinates on the keyboard, and a small blue square appears at the selected location. The user is prompted for a corner of the element. One corner of

4

the small blue square should be selected by the user with the mouse (using MB1). The selected corner is highlighted as a yellow dot, and then the user selects a position for the corner either by using the mouse or by typing the coordinates on the keyboard. This procedure is repeated for four corners. Once the first corner is selected, it is recommended that the remaining corners should be entered in a counterclockwise direction. The STUBBS program requires that the nodes be in a counterclockwise order. The XMESH program will automatically place the nodes in counterclockwise order regardless of how the nodes are entered by the user.

After the fourth corner is entered, the element appears in dark blue. The user can create as many elements as desired by repeating this procedure. When the user is done, the right mouse button (MB3) is selected and the elements are redrawn to fill the screen. At this point, all of the main menu items appear highlighted except for **Step**.

Load mesh. When **Load mesh** is selected, a dialog box appears that contains three buttons located along the bottom of the box, a selection line, a scrollable directory list, a scrollable file list, and a filter line. When the box is opened the cursor is located at the end of the selection line. The user may either type the filename <CR> to load the file, or the user can make a selection from the scrollable lists with the mouse. When a selection is made with the mouse, the file name is highlighted, and the user may confirm the selection by clicking on the Ok button. When Ok is selected, the mesh file is loaded into XMESH. If the user clicks on the Cancel button, the box is closed. A filter button is located between the Ok and Cancel buttons.

Once a file is loaded or created with **New**, all of the menu items under file are turned on except for **Animate Steps** and **Capture Steps**. If the user selects **New** after a mesh file has been loaded into XMESH, the user is prompted,

"NEW: Are you sure? (Y or [N]) _"

If the user replies with n <CR>, the prompt is erased and the user is free to continue with other menu items. If the reply is y <CR>, the drawing area is erased, the command line prompt is "Mesh filename: _", and the message line reads "Enter a fileneame for the new mesh."

Load Data. The **Load data** item operates like the **Load mesh** item. The filter is set to files with a "plt" extension. The plt files are one of the STUBBS output files. A mesh file must be loaded prior to the loading of a finite element data file. Once a data file is loaded, the user is ready for post processing. Further discussion is found in the chapter on post processing.

Directory. This item displays the dialog box produced by selecting **Load mesh**. The message line reads, "Displaying the Current Directory." The lists may be scrolled, but files can not be loaded with this feature.

Save. When selected the **Save** item prompts, "Mesh filename : _" The user types in a filename <CR>. If the file name corresponds to a name in the active directory, the user is prompted, "Overwrite?" to which the user should respond with (y)es or (n)o. If the user responds no, then the prompt asks once again for the file name. If the user responds yes, then the old file is overwritten with the new mesh data. If the filename is new, the mesh file is saved and XMESH remains open with the newly saved file as the current file.

Dump. **Dump** saves a portion of the drawing area as a image file. When **Dump** is selected, the user is prompted, "Text: -" and the message line reads, "Enter text to display with dump." When text is entered and terminated by pressing <CR>, a rectangular outline appears in the center of the drawing area. The mouse pointer is shown as an arrow which acts as a handle on the center of the rectangle. The prompt reads, "Position the text." The user can drag the rectangle across the drawing. When the rectangle placed in position by the user, the text can be placed on the drawing area by pressing any mouse button. Once positioned, the prompt reads, "Move text? (y or n)_" If the user replies y <CR>, the text is replaced by the rectangle and the user can again position the text. If the user replies n <CR>, the prompt reads, "Add more text? (y or n)_" A reply of y <CR> prompts the user for the text. A reply of n <CR> prompts the user, "Name of file: _" The user should enter a file name. Then the message line reads, "Select portion of screen to dump" and the mouse pointer is changed to represent the upper left corner of a rectangle. The user should move the mouse to the upper left corner of the area to be dumped and press *and continue to depress* MB1. Then the mouse pointer changes to become the bottom right corner of the rectangle. *The user should drag the mouse to the lower right corner of the area to be dumped and release MB1.* The screen will be written to a file and the user is free to continue working with the mesh file. The dumped file may be viewed using the X-windows undump command. More about viewing dumped files is found in the chapter on post processing under the **File/ Capture Steps** menu item.

Animate Steps / Capture Steps. Neither of these menu options is active for preprocessing. Both of the options are described in the chapter on post processing.

Exit. Section of **Exit** will terminate XMESH and close the XMESH window unless the user has made changes to the mesh file and has not saved the file. If elements were added or deleted, the user is prompted, "Mesh is altered. Want to Save? _" The user should reply y or n followed by < CR>. If the reply is n <CR>, the window is closed. If the reply is y <CR>, the user is prompted, "Mesh filename: _" The user enters the filename <CR>, the file is saved and the window is closed.

6

Edit menu

When the user selects the **Edit** menu item, a pull-down menu appears as follows:

> **Refine**
> **Redo**
> **Undo**
> **Align nodes**
> **Delete Elements**
> **Move node**
> **Update**
> **Elliptical nodes**
> **Smooth mesh**
> **Renode**

Refine. This item under **Edit** is very useful in building a complex, detailed mesh with little effort.

Refine subdivides elements into smaller elements. Before discussing the **Refine** item, it is necessary to discuss two features of the **Set** menu that control how **Refine** operates.

Redo and Undo. These items are discussed in detail later in this chapter.

Align nodes. The user is prompted to select first point of an alignment line using either pointing with the mouse (MB1) or coordinates entered via the keyboard. After selecting the first point, the user is prompted to select the second point. After setting the second point of the alignment line, the user is prompted to make a fence. Any nodes within the fence will be aligned to the previously selected line when the second (diagonal) corner of the fence is set. If the user is satisfied with the alignment, MB3 is selected. The user may now continue to pick a second alignment line, or select MB3 again to complete the alignment activity and release the mouse pointer to move out of the drawing area. If the user is not satisfied, the fence may be redrawn, or after selecting MB3, a second alignment line may be created.

Delete Elements to remove an existing solid, slip, or beam element. Bar elements can not be deleted without editing the mesh file. When selected, the user is prompted to set the corner of a fence using MB1. When the corners of the fence are defined, the elements whose centers are inside the fence will be shown in solid yellow to indicate that they have been selected. If the fence does not completely enclose the center of at least one element, the fence will disappear and the user remains in the **Delete Elements** menu item and the user may create another fence. If the user wishes to stop the **Delete Elements** process after elements have been identified by the solid yellow color, the user may select ESC from the keyboard. If the user selects escape, the yellow solid designation is removed and the elements are restored to their appearance as before the **Delete Elements** editing process was begun. Escape terminates the editing activity.

Move node. The user will be prompted to select a node with MB1. The selected node is then highlighted with a yellow dot and the user is prompted with "New location." The coordinates of the selected node will appear at the far right of the text line at the bottom of the screen (See Figure 1). When the location has been selected, the node will snap immediately to the new location. The command line will again prompt, "Select a node." The user may continue to move nodes or select MB3 to end the editing session.

Update. This item overwrites the current mesh file. The current file name is retained. No prompts or indications appear to the user other than a brief flash of the screen as the newly saved file is reloaded into XMESH. This feature should be used after major mesh changes such as mesh refinement or deletion of selected elements.

Elliptical Nodes. This item enables the user to place nodes on the surface of an elliptical curve. The user must input the two axis of the ellipse. When the item is selected, the user is prompted to select the first point on the X-axis. The user may use MB1 or type the coordinates. Then the user is prompted to "Select second point on X-axis." Then the user is prompted, "Select first point on Y-axis," then "Select second point on Y-axis." Then the user selects the nodes to fit to the ellipse by using one or more fences. When all the nodes have been selected, the user ends the process by pressing MB3 and the nodes are fitted to the ellipse.

Smooth Mesh. The user can smooth the mesh by selecting a number of nodes to remain fixed and then selecting the menu item. The user fixes selected nodes by using the **Set/Smoothing Codes** menu item. The usual sequence is to select **Show/Smoothing Codes**. The nodes are then color coded, red for fixed nodes and yellow for free nodes. Then use **Set/Smoothing Codes** to fix nodes as appropriate. Then use **Edit/Smooth Mesh** to move free nodes according to an interpolation scheme that attempts to make the distance between nodes more uniform.

Renode. This item causes the nodes to be renumbered and excess nodes deleted. This feature should be used if elements are changed from linear (no mid-side nodes) to quadratic (with mid-side nodes) or vice versa. The feature should also be used after elements are added or deleted.

It is advisable to use Update or save the file after using renode, and then, reload the file to ensure that the proper node accounting and element connectivity is carried from one editing activity to the next.

The **Set** menu items are shown below.

Block
Boundary conditions
Element type
Material
Refine
Smoothing Code
Unset Smoothing Code

Snap size

Only the items **Set/Refine** and **Set/Block** are discussed at this time to allow a complete description of the capabilities of **Edit/Refine.** The remainder of the **Set** menu items are discussed later in the section on the **Set** menu.

When the user selects **Set/Refine** the user is prompted with:

1- keep tria 2- avoid tria 3 -avoid all tria

The word "tria" refers to triangular elements. In STUBBS and other finite element programs, triangular elements have some poor computational qualities as compared to rectangular elements. Triangular elements can be useful as transition elements when changing the density of the mesh. However, triangles should be avoided at locations of particular interest to the analysis. The **Set/Refine** item allows the user to specify if triangles may be used in subdividing elements. In many cases, triangular elements are impossible to avoid and will be created during the refining process regardless of the selection made under **Set/Refine**.

The **Set/Block** feature allows the user to control the subdivision of elements by either selecting or blocking the vertical and horizontal directions in the refining process. If neither the vertical or horizontal direction are blocked, then a rectangular element is subdivided into quarters. If either the vertical or horizontal direction is blocked, then the element is divided into two halves. If both the vertical and horizontal directions are blocked, then no refinement of the element is permitted.

When **Set/Block** is selected a dialog box appears in the middle of the drawing area. The dialog box is titled "Set Blocking Codes." Within the dialog box is a hollow rectangle, whose sides can be toggled from transparent to black by MB1. When the side is transparent, the side may be divided during refining. Initially, the blocking codes are defaulted to transparent. If the side is switched (toggled) to black by the user, the side may not be divided during refining. At the bottom of the dialog box are buttons to select either **Ok** or **Cancel**. When **Ok** has been selected, the user is prompted to choose the corners of a fence. All sides within the fence will be assigned the selected blocking codes. By using combinations of blocking codes, a mesh can be refined in selected zones to any level desired by the user.

Once **Set/Refine** and **Set/Block** have been selected, the user may have the mesh automatically refined by selecting **Edit/Refine**. Three related menu choices are used in refining, **Edit/Refine**, **Edit/Undo**, and **Edit/Redo**.

Selection of **Edit/Refine** quarters or halves elements and is controlled by the **Set/Refine** and **Set/Block** options. Repeated selection of **Edit/Refine** further refines the mesh. **Edit/Undo** returns the mesh to its arrangement prior to the last selection of **Edit/Refine**. **Edit/Redo** re-establishes the mesh arrangement that existed prior to the last **Edit/Undo**. The Undo and Redo selections are limited to the reversing and reestablishing the refinement done

using Refine and Undo respectively. Undo and Redo can not operate on elements not previously manipulated under the Refine menu selection in that editing session. **Caution:** Use of the **Edit/Refine** feature will convert beam elements to solid elements and destroy slip elements.

Set Menu

The **Set** menu has been described in part to explain the blocking and refining settings. The complete sub menu of set appears as:

Block...
Boundary Conditions
Element type ▸
Material ..
Refine
Smoothing Code
Unset Smoothing Code
Snap size

Boundary Conditions. This option allows the user to either fix or free the displacement and pore water degrees of freedom for any node. When **Set/Boundary conditions** item is chosen, the user is prompted with:

Set (X), (Y), or (P)wp_

The user must select either X, Y, or P.

Then the user is prompted with:

Set to f(I)xed or f(R)ee?

The user must select I or R.

Then the user is prompted with:

MB1-set corner MB2 - start over MB3-exit

The user should make a fence that contains the nodes that are to have their boundary conditions altered. The user should continue to fence nodes until satisfied, and then select MB3. No visual clues that the boundary conditions have been altered are visible unless the boundary conditions indicators have been turned on using.

Show/Boundary conditions. The indicators are visible when a boundary condition has been fixed to some constant value. A pair of red arrows bracketing the node is used for the displacement degrees of freedom and a red dot for the pore water degree of freedom. Absence of these indicators when **Show/Boundary conditions** is turned on, indicates that the node is free to take on any value that satisfies equilibrium.

10

Set/Element type. This item permits the user to specify the type of element that is to be created either by the **Make** main menu item or by the **Edit/Refine** process. When the **Set/Element type** is selected a two-button submenu pops out that reads, **Linear** on one button and **Quadratic** on the other button. Linear elements have corner nodes and less than a complete set of mid-side nodes. Three-node triangular elements and four-node rectangular elements are linear elements. Six-node triangular elements and eight-node rectangular elements are quadratic elements. After the user makes a selection of element type, the user is prompted to make a fence, and when the fencing is concluded, the selected elements are converted to the appropriate type.

Set/Material. This item permits the user to assign a material number to an element or set of elements. When selected, the user is prompted for a material number. The user should enter an integer. If the material name has not previously been created, the user will be prompted for a material name. Then the user will be given a fence prompt. Each material is assigned an individual pattern that will be displayed if the **Show/Material** item is turned on. The pattern can be shown in a legend that is placed in the upper right corner of the drawing when the **Legend/Material/Elements** menu item is selected.

Set/Refine. This menu operation was previously described in the section on **Edit/Refine**.

Set/Smoothing Code and Unset Smoothing Code. These options are used to fix and free nodes during the Smoothing operation. The Set/Smoothing Code item fixes the position of the nodes.

Set/Unset Smoothing Codes. This item frees the position of the nodes and make them subject to repositioning during smoothing.

Set/Snap size. This menu item is another of the CAD-like features of the program. The drawing area is described by a coordinate sets that compose the lower-left corner and the upper-right corner. Within the drawing area, the mouse pointer can snap to coordinates that are found on a specified interval within the drawing area. For example, let us assume that the lower-left corner of the drawing area is 0, 0 and that the upper right corner is 10, 10. If snap is set to 0.1, the drawing area will be subdivided into coordinates with an X- and Y- spacing of the snap size, 0.1. If snap is turned on by selecting **View/Snap** and the mouse pointer is moved across the drawing, the pointer jumps from one snap coordinate to the next. If snap is turned off, the mouse pointer will move smoothly across the drawing area in a continuous motion. Turning snap on and setting a convenient snap size can make the creation of new nodes at precise points within the drawing area an easy operation.

Show Menu

The **Show** menu turns on visual indicators that can assist the user in understanding the status of various settings and conditions. In fact, the display of indicators is vital to verifying the current status of these codes and conditions. The operation of the **Show/Boundary conditions** indicators was explained in the section on **Set/Boundary conditions**.

The complete **Show** menu is presented in Table 1.

Table 1 Items Listed for SHOW Menu	
Bars	Toggles to display/hide bar elements
Slip	Toggles to display/hide slip-separation elements
Boundary conditions	Toggles to display/hide boundary condition code indicators
Element centers	Toggles to display/hide element centers
Element numbers	Toggles to display/hide element numbers
Element types	Toggles to display/hide four-node solid elements
Grid	Toggles to display/hide element boundary lines
Materials	Toggles to display/hide material type
Node locations	Toggles to display/hide node locations
Node numbers	Toggles to display/hide node numbers
Title	Toggles to display/hide a file name in the upper left corner of the drawing area
Smooth codes	Toggles to display/hide smoothing codes
Curve codes	Toggles to display/hide curve codes
Scale ▸	Presents a submenu to place X- and Y-scales on the drawing area and move scale the scales, once displayed
Block	Toggles to display/hide blocking codes for element sides
Color contours Displacements Tensors Vectors	NOT ACTIVE with mesh file Active with plot file for post processing

View Menu

The **View** menu is the most CAD-like of all the menu items. The **View** submenu items are listed in Table 2.

Table 2
Items Listed for VIEW Menu

Fit	Redraws the mesh to fill the window.
Pan	The user is prompted to indicate a new center for the viewing area. This feature enables the user to move the viewing area to portions of the drawing area that are not visible in the current window. This feature is often used after zooming.
Zoom	The user can zoom in on the mesh. A rectangular outline appears in the center of the drawing area. The rectangle has a handle in the center and can be moved about the window. The user can increase or decrease the size of the rectangle by pulling on any corner. When the rectangle includes the area of interest the user should select MB1 and the mesh appearing within the rectangle is redrawn to fill the drawing area.
Snap	Toggles on grid snap. The cursor will jump to points located at regular intervals set by the Set / Snap size menu item.
Shrink	Shrinks the extent of each element while keep in the center of each element fixed. The user can better see the relative location of elements within the mesh.
Redraw	Redraws the image. Draws mesh from the data.
Refresh	Redraws the image. Recalls image from memory. Preferred choice for redrawing a complex mesh.
Full	Resizes the drawing area to fill the available space within the border.
Move	Moves the location of the drawing area on the screen as the border remains the original size.
Resize/ move	Moves and resizes the drawing area as the border remains the original size.

Make Menu

The **Make** menu item allows the user to create individual elements of any kind: solid, beam, bar, and slip-separation. The sub menu items are listed in Table 3.

Table 3
Items Listed for MAKE Menu

Solid	Makes solid element
Beam	Makes beam element
Slip	Makes (horizontal) slip element
Double Slip	Makes double slip element - one slip on one slip
Inclined Slip	Makes (vertical) slip element
Inclined Double Slip	Makes (vertical) double slip element
Anchor bar	Makes 2-node bar element
Reinforcement bar	Makes 3-node bar element

When elements are created by **Make/Solid**, it is recommended that the nodes be entered in a counterclockwise direction. STUBBS requires that the nodes be entered counterclockwise in the element connection tables. XMESH will automatically place the nodes in counterclockwise order. If a file in which the nodes are not in clockwise order were loaded into XMESH, the program does not change the order and may create new elements using **Edit/Refine** whose nodes may not be ordered in a counterclockwise direction.

Make/Solid and **Make/Beam**. The user is prompted for the approximate center of the element. The user selects the position with the mouse or by typing coordinates using the keyboard, and a small blue square appears at the selected location. The user is prompted for the corner of the element. One corner of the small blue square should be selected by the user with the mouse (using MB1). The selected corner is highlighted as a yellow dot and the user then selects a position for the corner using the mouse or types coordinates on the keyboard. This procedure is repeated for four corners. When the fourth corner is entered, the element appears in dark blue. The user can place as many elements as desired by repeating this procedure. When the user is done, the right mouse button (MB3) is selected and the elements are redrawn to fill the screen. Beam elements are six-node elements. The three-node sides are placed on the longer sides of the element.

Make (for Bars and Slips). Bar element and slip elements are two or three node elements. The bar or slip elements are created by identifying existing nodes that are corner nodes of solid elements. Slips elements, both single and double, are created by using a fence. Inclined slip elements, both single and double, are created by using MB1 to identify nodes. Bar elements are created by using MB1 to identify corner nodes. Bar elements are either anchor bars or reinforcing bars. Anchor bars are two node elements that can be attached to any two nodes. Reinforcing bars must be attached to the side of a solid element. Reinforcing bars may not span elements; anchor bars may span any number of elements or may form pieces of a truss or frame. *If solid elements are deleted whose nodes are part of bars or slips, the bar and slips are also deleted. It is advisable to create, refine and delete solids element before adding any bars, slip elements, or beam elements. Use of **Edit** functions may result in unpredictable consequences when beams, bars, or slips are part of the mesh.*

Size Menu

The **size** menu has five items, but only **Bar** and **Gap** are active during preprocessing. The **Bar** item is used to select the line thickness to be used when bars elements are visible within the drawing area. In order for Bar elements to be visible, first, they must have been created under the **Make/Anchor bar** or **Make/Reinforcement bar** menu items, and second, the **Show/Bars** menu item must be turned on. When **Size/Bar...** is selected, a dialog box appears that contains five horizontal bar segments stacked into a column. The bar segments have thicknesses that grade from the thin top-most bar to the very thick bottom bar. The user selects the bar segment

whose thickness suits his needs and clicks the "Ok" button in the dialog box to have the bars in the mesh redrawn to the selected thickness.

The other active menu item is **Gap**. The **Size/Gap** feature is used with the **View/Shrink** feature. When **View/Shrink** is turned on, each element is drawn at a slightly reduced scale so that space appears between each element. When shrink is used, each element can be seen as a separate body. The magnitude of shrink can be selected by using **Size/Gap**. First, turn on **View/Shrink**. Then, select **Size/Gap**. A dialog box will appear. The dialog box title is Gap Size (percent). Within the box are a column of magnification/reduction factors that are dimensioned in percent. The factors range from 10 to 1,000. As the user selects a factor, the elements are redrawn real-time in response. The user may select any combination of factors that will provide an acceptable shrink size. Once the user is satisfied, an "Ok" button located to the right of the factors should be selected to close the dialog box. The user may select the "Cancel" button to close the dialog box and restore the size of the elements to the size in effect before the **Size** menu item was selected.

Legends Menu

The **Legends** menu contains the items **Material** ▸ and **Color**. Color is inactive during preprocessing. The **Material** menu has choices of **Elements** or **Bar**. The **Legends/Material** item displays a small box showing the all of the material (solids or beam) that are have been specified by the user during the creation of elements. The **Legends/Bar** item displays a small box showing all of the bar element materials that have been specified by the user in the creation of bars.

Batch Menu

The **Batch** menu allows the user to activate several menu choices at one time. Without **Batch**, the user must make a single menu choice and have the choice executed, and then make additional choices and have these executed, one at a time. When the user selects **Batch**, the menu title toggles to **Enter**. The user can then make several menu selections which are then listed below **Enter**. When the user has made all of the desired selections, he may select **Enter** and all of the chosen menu items will be executed at one time.

For example, the user could turn on boundary conditions, node numbers, and materials by making the choices listed in Table 4.

Table 4 Possible Combination of Selections Using BATCH Menu	
User selection	Screen action
Show/Boundary conditions	Boundary condition indicators are turned on
Show/Node numbers	Node numbers are turned on
Show/Materials	Material hatching is turned on

Alternatively, the user could process a set of commands using Batch as shown in Table 5.

Table 5 A Second Set of Possible Combination of Selections Using BATCH Menu	
User selection	Screen action
Batch	"Batch" label toggles to "Enter"
Show/Boundary conditions	"Show Boundary conditions" appears below "Enter"
Show/Node numbers	"Show Node numbers" appears below "Show Boundary Conditions"
Show/Materials	"Show Materials" appears below "Show Node numbers"
Enter	Boundary condition indicators are turned on AND Node numbers are turned on AND Material hatching is turned on AND "Enter" toggles back to "Batch"

Build Menu

The **Build** menu allows the user to build an input file for STUBBS. **Build** should not be selected until the mesh has been finished and no further alterations are planned. The selection of **Build** marks the end of mesh preprocessing. The **Build** menu is described in the Chapter 3 on input file generation.

3   Input File Generation

A mesh file contains only node data and element connectivity data. The mesh data are only part of the information needed to perform a finite element analysis. Material properties, declaration of problem type, and instructions regarding construction activities are needed to complete the information needed by STUBBS. The **Build** menu is the feature that creates input files that can be used directly by STUBBS.

The **build menu** contains fifteen menu items. Three menu items are active with a mesh file loaded. These items are listed in Table 6.

The **Build** menu items are added to the input file in the order in which they are selected. Some care must be taken to ensure that portions of the input file are in appropriate order. For example, material properties must be placed in the data file before the start of the construction script.

Build/Initialize prompts the user Enter the name of the stubbs file to create: _ while the message line reads "Building a construction Script."

When a file name is entered the prompt changes to Enter the file to store time data:_

After entering a file name a large dialog box appears. The user enters the title line, information for KW SPEC, information for KW UNIT (default is 14.78 Patm, 0.0361 GAMMAW, 1.0 for g), and five comment lines.

When the data entry is terminated by pressing the "Done" button with MB1. The line reads "Initialization Completed..." When MB3 is clicked, the message is erased.

After initialization, the **Initialize** and **Continue** items are de-activated and all of the remaining menu items are activated except for Const_Script.

It is recommended that the remaining data be completed by selecting **Pre-existing, Matl_Properties, Time_steps**, any other items, and end with **Write_Output**.

Table 6	
Items Listed for BUILD Menu	
Initialize	Prompts for two file names: data file and construction script file
Continue	Prompts for existing data file in order to append new data lines during the current session
Review▸	Prompts for file containing construction script data.
Pre-Existing▸	Creates KW XELT (solids), KW XBRE data (bars), KW XBND (beams), or KW XSLP (slips)
Matl_Properties▸	Creates KW SMPR (solid), KW BRMP data (bar), KW SLMP(slip), or KW BMMP (beam)
Time_Steps▸	Creates KW TIME/ KW DONE data
Const_Script▸	Creates one of eight possible KW's depending on sub menu selections KW CSFP/CSEX data (adds/removes solid elements), KW CSFL/CSFP (adds/removes beam elements), KW CSSP/CSSR (adds/removes slip elements), KW CSBP/CSBR (adds/removes bar elements)
Output_Control▸	Creates KW VBOF/VBON and KW RPOF/RPON
Add_Comment	Adds a one line comment to the input file
KW CSBC	Creates KW CSBC data (changes boundary conditions)
KW IDPP	Creates KW IDPP data (sets initial conditions for pore water pressure and displacements)
KW CSBL	Creates KW CSBL data (applies a distributed load to the side of an element
KW CSPL	Creates KW CSPL data (applies a concentrated loading to a node)
KW ENDO	Creates KW ENDO data (material properties for the enchronic plasticity constitutive rules)
Write_Output	Writes input record and construction time steps to files named in Initialize.

Build/Preexisting. A sub menu of the four element types appears. Solid, Bar, Slip, and Beam. When solid is selected, the user is prompted to select elements by making one or more fences. When the fence process is terminated by pressing MB3, the message line reads "Solid elements selected are stored." When MB3 is pressed once again, the message line is erased and the **Build** pull-down menu is removed.

Build/Matl_Properties. A submenu of the four element types appears: Solid, Bar, Slip, Beam. If solid is selected, a dialog box appears that is titled Xmesh (Build) - Table for Existing Solid Material Properties.

TITLE GAMMAT RVOID GS Y.MOD P1 P2 XN XM K0 Kx Ky
B VGA1 VG A2

MATL1 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00
0.00 0.00 0.00

The **Build** option automatically provides as many lines for solid material types as were assigned to elements in XMESH. The **Build** menu does not allow you to add materials that were not assigned during preprocessing. The process is terminated by hitting the "Done" button. The **Build** pull-down menu is removed. The feature will provide an appropriate dialog box for all material types.

Build/Time_Steps. This item provides a sub menu of **Start**, End, and **Repeat**. **Start** and **Repeat** are active; End is not active. Pressing **Start** produces a prompt "Time length:" and a message reading "Enter the Length of Time." The user enters a number.and the prompt becomes: "Number of Sub Steps:" and the message reads "Enter the Number of Substeps." When the value is entered, the prompt disappears and the message reads, "Total Time = 1.000 Verbose = ON and Report = ON."

After **Build/Time_Steps/Start** is selected, the **Start** and **Repeat** items are made inactive and **END** is made active. After **End** is pressed, a concluding KW DONE is added to the data lines and the **Start** and **Repeat** items are re-activated and the **End** item is again turned off, the pull-down menu is removed.

The **Build/Time_Steps/Repeat** produces a prompt "Time length:" and a message reading "Enter the Length of Time." After the user enters a number the prompt becomes: "Number of Sub Steps:" and the message reads "Enter the Number of Substeps." After the value is entered, the prompt disappears and the message reads, "Total Time = 1.000 Verbose = ON and Report = ON." After the value is entered, the prompt becomes, "Number of Repetitions and the message becomes," Enter the Repetitions." After the value is entered, the prompt is erased and the message reads, "Total Time = 14.000 Verbose = ON and Report = ON." The total time value is calculated by XMESH.

Build/Constr_Script. This menu item is activated only during time steps. When selected a submenu pops up with "Solids Bars, Slip, and Beam. The user selects one of the element types and is then queried "(E)xcavation or (C)onstruction:" The user enters either an E or C. Then the user is asked to select the element with the mouse. The user encloses the elements within one or more fences and then selects MB3 to complete the selection process. The appropriate KW is added to the data file followed by the element numbers affected.

Build/Output Control. This item allows the user to add KW's Verbose OFF and Report Off to the data file. The selection initially turns the selected output control off using VBOF or RPOF. Repeated selection of these items toggles the control ON and OFF. The current status of the controls is shown in the message line of the screen when the output control item is selected.

Build/Add_Comment. This menu item prompts the user to add a one-line comment to the data file by typing the comment into the command line portion of the screen.

Build/KW. These items provide either dialog window templates (KW ENDO, KW CSBC) or prompts to select nodes (KW CSBL, KW CSPL) or prompts to set the total PWP for all the nodes to a selected value of total head (KW IDPP).

Build/Write_Output. This choice writes a final KW DONE into the output file that was named during the response to the **Build/Initialize** selection. The Build menu items are added to the data file in the order that they are selected, so **Build/Write_Output** should be the final selection chosen.

4 Postprocessing

XMESH has powerful contouring and plotting capabilities that enables the user to interpret the results of an analysis. These capabilities can be used to examine any appropriately structured data set but the program was written specifically to work with the plot file produced by STUBBS. This chapter addresses the post processing of analysis performed with STUBBS. The generic use of XMESH is discussed in Chapter 5.

Two steps are required to perform post processing. First, the problem mesh must be loaded using the **File/Load mesh** menu item. Then, the STUBBS output is loaded by selecting the **File/Load data** menu item. Once selected, a dialog box appears that allows the user to select a *.plt file. The plot file (with a plt extension) is produced by STUBBS.

After a data file is loaded into XMESH, several main menu selections are disabled and others are enabled. The **Edit**, **Make**, **Set**, and **Build** selections are turned off. Under **Show**, the features **Color contours**, **Displacements**, **Tensors**, and **Vectors** are activated. The **Show/Block** feature is turned off. The **Size** and **Step** menus are activated. The **Display**, **Legends**, and **Batch** menus appear as they did during preprocessing.

File Menu

The **File\Load data** item loads a plot file that contains displacement data for nodes and stress data for elements. The contents and structure of the plot file are discussed in the Chapter 5 on generic use and also is discussed in a companion manual for STUBBS.

This menu item performs in a manner similar to that of **Load mesh**. When **Load data** is selected, a dialog box appears that contains three buttons located along the bottom of the box, a selection line, a scrollable directory list, a scrollable file list, and a filter line. When the box is opened the cursor is located at the end of the selection line. The user may either type the file-name <CR> to load the file, or the user can make a selection from the scrollable lists with the mouse. When a selection is made with the mouse, the file name is highlighted after the mouse pointer is placed on the file name, and the user may confirm the selection by clicking on the Ok button.

The user may chose to type in the name of the file rather than use the mouse. When Ok is selected, the mesh file is loaded into XMESH. If the user clicks on the Cancel button, the box is closed. A filter button is located between the Ok and Cancel buttons.

Once a plot file is loaded, a number of post processing menu items are activated and other items associated with pre-processing are inactivated. For example, the entire **Edit** menu is inactive after data is loaded. The **Step** menu item is activated and the **File\Animate Steps** and **File\Capture Steps** are activated.

If the user attempts to load a data file that is not compatible (in terms of the number of nodes and number of elements) with the currently loaded mesh, an error message appears in the message block of the window bottom panel that states that the selected file is not a data file for the current mesh file. A sample message is, ' **"apr17.plt" is not a data file for footing4.msh**.'

File\Animate Steps. This item displays the results of the analysis by drawing the data for selected groups of time steps in sequence. The automatic display of the steps gives the impression of animation. After the item is selected, the user is prompted,

"Enter the number of time steps:"

The user should respond by entering a number from the keyboard. The steps are displayed with about 2 frames per second. The last step remains shown in the window. The animation need not start at the first time step. The user may use **Step** to display the results of calculations for each time step. For example, the plot file may contain 8 timesteps. The user may use **Step** repeatedly to display timesteps 1, 2 and 3. Then the user uses **File Animate Steps** and enters 3 at the prompt. The next 3 steps (numbers 4, 5, and 6) would be animated and the results for step 6 would remain on the screen. If the user again selects **File\Animate Steps** and 3, the results of steps 7, 8 and 1 are displayed and the result of time step 1 remains on the screen. The **Step** menu item also loops back to the first timestep went the last time step in the plot file has been displayed.

File\Capture Steps. This item performs a function similar to the **File\ Animate Steps** plus the item dumps graphics images from each time steps to files, one file for each step. The item provides prompts similar to **File\ Dump**.

When **File\Capture Steps** is selected, the user is prompted, "Text: -" and the message line reads, "Enter text to display with dump." When text is entered and terminated by pressing < CR >, a rectangular outline appears in the center of the drawing area. The mouse pointer is shown as an arrow which acts as a handle on the center of the rectangle. The prompt reads, "Position the text.." The user can drag the rectangle across the drawing. When the rectangle placed in position by the user, the text can be placed on the drawing area by pressing any mouse button. Once positioned, the prompt

22

reads, "Move text? (y or n)_" If the user replies y <CR>, the text is replaced by the rectangle and the user can again position the text. If the user replies n <CR>, the prompt reads, "Add more text? (y or n)_" A reply of y <CR> prompts the user for the text. A reply of n <CR> prompts the user, "Name of file: _" The user should enter a four-character file name. Assume the user enters "BASE" as the four-character string. Then the message line reads, "Select portion of screen to dump" and the mouse pointer is changed to an upper left corner of a rectangle. The user should move the mouse to an upper left of area to be dumped and press *and continue to depress* MB1. Then the mouse pointer changed to a bottom right corner. *The user should drag the mouse to the lower right corner of the area to be dumped and release MB1.* The image contained within the rectangle will be written to a file named BASE00n.xwd for each step. The file name is composed of three part, the four-character string (base), the number of the step dumped (003, for the third step dumped), and the extension "xwd."

The number of the steps dumped is not the time step number, but the number of steps captured. For example, the user may have a file containing 20 time steps. The user may use the **Step** menu to advance to time step 4 and then use **File\Capture Steps** to dump the next 6 time steps. If the user entered the four-character string "hasp" for the file name, the fifth captured step is saved as file "hasp005.xwd." The contents of the file are time step 9 (step 4 + 5 additional step dumps). To view the dump file, the user should open another shell. From the UNIX prompt, in the newly opened shell the user may type the command 'xwud -in hasp005.xwd' to view the dumped image. Only one image may be viewed at one time.

Display Menu

The **Display** menu item is the heart of the post-processing. The five menu items under **Display** are: **Color...**, **Data...**, **Smooth contours**, **Time History**, and **Profile**. Of these five items, **Data...** is the primary post processing tool. The Display menu items are discussed in the order of **Data..**, **Color...**, **Smooth Contours**, **Time History**, and **Profile**.

When **Data...** is selected, a dialog box appears in the center of the drawing area. The box contains fifteen buttons arranged into three columns of information. The left column is headed by a button that toggles between "No conversion" and "Element to node." Below the "No Conversion" button is a title "Element Data" that stands above eight buttons that are used to select parameters that were calculated for solid elements during each time step in STUBBS. The eight parameters are listed in the following Table 7. The units for the parameters were set in the STUBBS input file using keyword UNIT.

Table 7 Element Parameters in XMESH Plot File	
Q	The maximum shear stress (radius of the Mohr's circle)
P	The average normal stress (the normal stress indicated by the center of the Mohr's circle)
Q/P	The ratio of Q to P
SIGMAXX	The horizontal (X-axis) normal stress
SIGMAYY	The vertical (Y-axis) normal stress
Tau_XY	The shear stress on the YZ plane
PWP	The pore water pressure
F	Not applicable to general users - used for percentage of mobilized shear strength for a specific project.

At the bottom of the left column is a time button that indicates the elapsed time for which the data was calculated. The time begins at 0.00 and is accumulated from the time step durations specified in KW TIME. When the time button is selected, the time is incremented to the end of the next time step. After the end of the last time step, the time returns to the initial value 0.00. The units of time are set in STUBBS in KW SMPR.

The center column shows a vertical color bar divided into eleven sections. Above the color bar is a button that shows the maximum value of the parameter that was selected from either the element data parameters listed in the left column or the node pore water pressure selected from the "PWP" button in the right column. Below the color bar is a button that shows the minimum value of the selected parameter. Either or both of the buttons may be reset to a user-specified maximum or minimum value by selecting the button and typing the selected maximum or minimum value into the button from the keyboard.

The right column contains three buttons. The top button toggles from a smooth continuum of colors to incremented bands of color. When toggled, the button does not change appearance, but the color bar is changed from the eleven boxes of colors to a smoothed color continuum and vice versa. The second button is located below a title, "Node Data," that stands opposite the "Element Data" title of the left column. The only button under Node Data is a "PWP" button that is used to select pore water pressure as a plotted parameter. A "Done" button is located at the bottom of the right column. After the "Done" button is selected, the dialog box is closed and the user sees the results of his selections plotted within the drawing area.

The **Color...** choice displays a color bar and a color triangle. The color bar is a horizontal bar divided into eleven sections. A cross appears in the center of the section located at the left end of the color bar. Above the color bar is a scale shown in white. The title of the scale is shown in white above the center of the scale. The title for the scale is determined by the parameter selected in **Display/Data..** (discussed in the previous section)

24

The color triangle is shown below the color bar. The top vertex of the triangle is red, the lower-right is green, and the lower-left is blue. The sides of the triangle are divided into 10 sections and the interior shows 100 combinations of red-green-blue (RGB) hues. Each of the eleven color blocks in the color bar may be give any of the hues from the 100 selections within the color triangle. When one of the small triangles within the color triangle is selected with MB1, the hue is assigned to the section of the color bar that contained the cross, and the cross is moved one box to the right on the color bar. When a color is selected for the right-most box, the cross moves to the left-most box. The cross can be moved to any of the boxes by selecting the box with MB1. When the colors assigned to the blocks in the color bar suit the needs of the user, the user should select the "Done" button that is located to the right of the top vertex of the color triangle.

The ability to select colors gives the user the opportunity to customize the appearance of the data contours. Sometimes the user will want to set several of the boxes in the color bar to the same color. For example, the user may wish to show all values less than zero as the same color. With all of the color boxes showing values below zero set to the same color, the zero zones in the plot will be clearly displayed after the user selects the done button. Another potential application is to set all values higher than some certain value, say some strength value, to the same color to indicate zone of potential failure.

The **Display/Smooth contours** menu item toggles from showing contours of parameters values to a smoothed color continuum formed by interpolated values for the parameters. This menu item performs the same function as the Smooth contour button in the **Display/Data...** dialog box.

The **Display/Time History** menu item provide a means to plot parameters that were calculated for a particular location within the mesh. When the user selects the **Display/Time History** item, the user is prompted to "Select a point - MB1 (MB3 to finish)." When the user selects a point with the mouse, the nearest element to the mouse pointer is displayed in yellow and the user may complete the selection by pressing MB3. Immediately following the selection of MB3, a new plot window is created on top of the drawing area.

The new plot window has a border with a title block reading "Time History Plots." A menu bar contains four items: **File, X-Axis Data, Y-Axis Data,** and **Change Parameters.** Under the **File** menu are selections of **Print, Clear, Export,** and **Quit. Print** sends the image of the current drawing area to a printer. **Clear** erases the drawing area. **Export** creates a ASCII file of the current X-Axis and Y-Axis parameters in the directory that holds XMESH. **Quit** closes the plot window.

The user can plot a wide variety of parameters and control the use of grid lines, tick marks, and scales. A word of caution should be delivered regarding elements that are excavated during the course of the analysis. After an element is excavated, data for the element and its nodes continue to be written to the plot file for all time steps. Point (or nodal) data (displacement, for

example) on the Time History plot is interpolated from data of the four corner nodes that compose the element. Element data (stress, for example) is taken directly from the element data in the plot file.

If an element were excavated, the stresses shown in the plot file are zero. When an element is excavated (or not yet placed as fill), no data is plotted using the **Display/Time History** menu item for the period when the point being plotted is within an excavated element. That is, the element does not exist during certain times. If a point being plotted is within an element that is excavated and then later restored as fill, the data for the fill is plotted using a dotted line. A print file may be obtained by selecting the **File/Print** menu item in the *plot window*. The naming convention for a print file of time history data is the mesh file name followed by prn. For example, if the mesh file were sep20.msh, the print file is called sep20.msh.prn (note: unix file name). A sample of a time history plot print out is shown as Figure 2.

Profiles may be plotted using the **Display/Profile** menu option. Similar plotting choices are available to plot profiles as are available using the **Display/Time History** menu item. Time History plots data based on a selected point. Profile plot data are based on a line of nodes. The line of nodes may be either vertical of horizontal. The line does not have to be a straight line, but only a notional line. After the **Display/Profile** item is selected, the user is prompted to select either: "(H)orizontal or (V)ertical." If horizontal were selected, the data is plotted with the X-axis using the selected nodes X coordinates. Another variable may be plotted on the Y axis. If vertical were selected, the data is plotted with the Y-axis using the selected nodes Y coordinates. Another variable may be plotted on the X-axis. The variables available for plotting are similar for both menu selections, **Display/Time History** and **Display/Profile**. The list of variables shown in this chapter is typical of plot file data. The particular selection of variables used depends on the contents of the plot file.

Displacement, both X and Y, Time, Sigma XX, Sigma YY, and Tau XY will be found in any STUBBS plot file. Pore water pressure (PWP), and F are additional variables that are often placed in plot files. The variables Q and P and the ratio Q/P are produced by manipulation of the stress data by XMESH.

A profile plot produced by selecting the **Display/Profile** menu item will not plot data for sections of the profile that contain elements that either have been excavated or are yet to be placed. The profile plot is a series of lines drawn from node location to node location. The node location appears on the computer screen as a red dot. When the an element is removed for an interval of the profile, the node symbol for a node either in the interval or terminating the interval are shown as a blue dot. A print file may be obtained by selecting the **File/Print** menu item in the *plot window*. The naming convention for a print file of profile data is the plot file name followed by prn. For example, if the plot file produced by STUBBS were sep20.plt, the print file
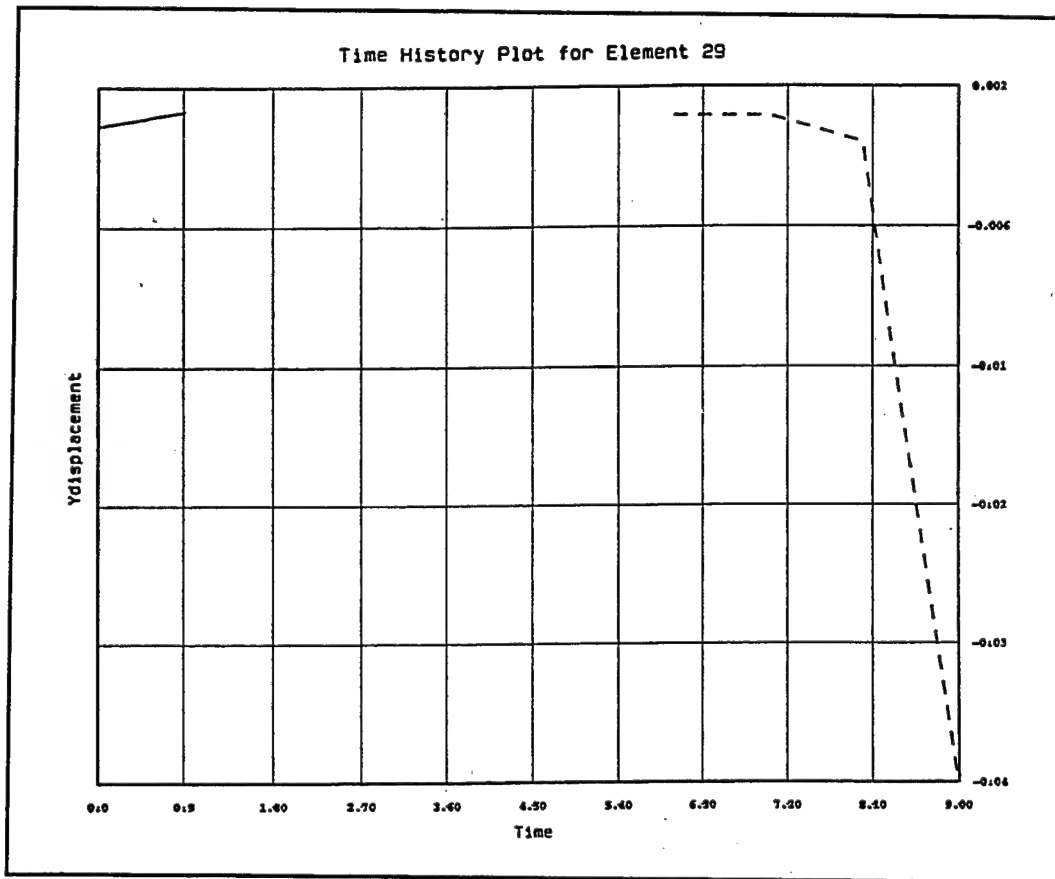
Figure 2. Sample time history plot

is called sep20.plt.prn (note: unix file name).   If the profile is printed, a blue dot is shown as a solid circle and a red dot is shown as an X symbol. A sample of a profile plot print out is shown as Figure 3.

The selections under the **X-Axis Data** and **Y-Axis Data** are identical. The choices under each item are listed in Table 8. The definitions of the parameters were shown earlier in Table 7. The selection from the **Change Parameters** submenu are presented in Table 9. The plot window has all of the capabilities of any window, in that it may be re-sized by using the mouse to grab a handle point on the window border, or minimized to an icon by clicking in the minimize button on the menu bar. The window can be restored to full size by clicking on the icon.

Show Menu

The **Show** menu item has some special capabilities during postprocessing. The menu item **Size** is used in coordination with **Show**. The **Size** item has

items of **Tensor, Vector,** and **Displacement**. The definitions of the **Show** menu items are provided in the Table 10.
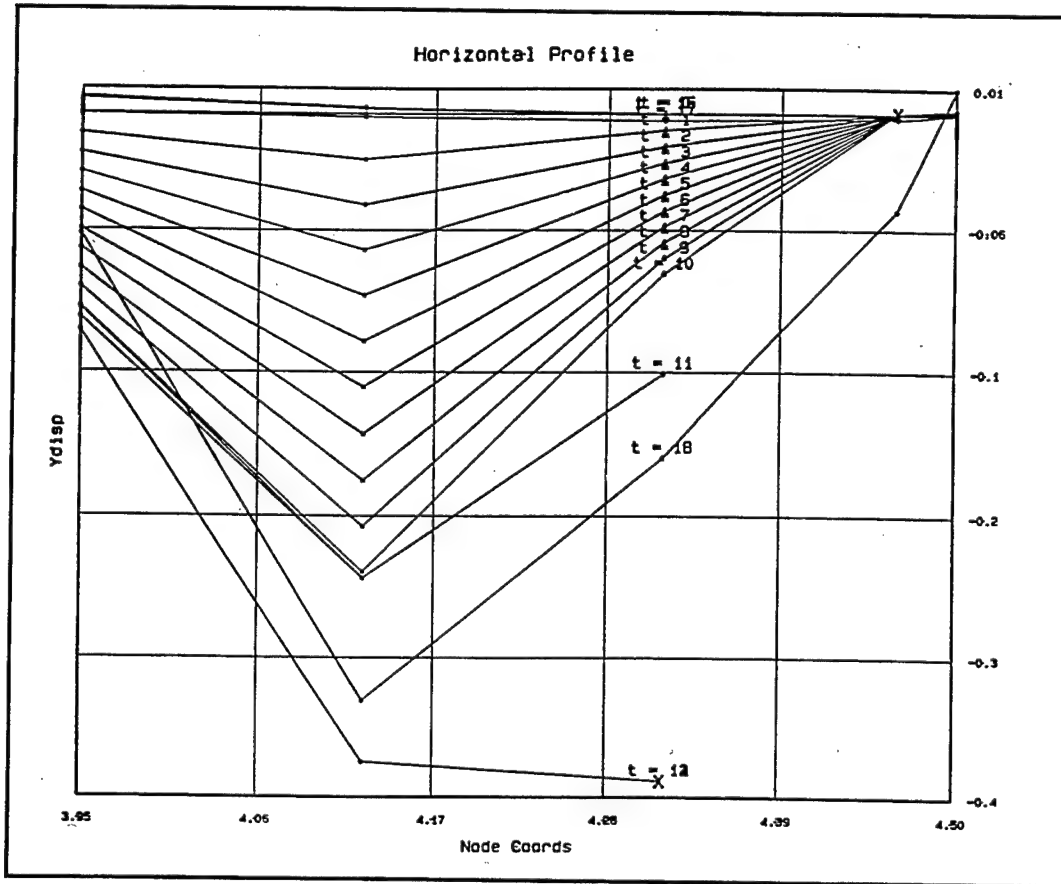
Figure 3.   Sample profile plot printout

Table 8 Selections for DISPLAY/PROFILE Menu	
PWP	computed (averaged) at the nodes
PWP	computed (averaged) at element center
F	not applicable - reserved for future
Xdisp	computed at element center
Ydisp	computed at element center
Time	total elapsed time
Sigma XX	computed at element center
Sigma YY	computed at element center
Tau XY	computed at element center
Q	computed at element center
P	computed at element center
Q/P	computed at element center
Log time	base 10 logarithm of elapsed time

Table 9
Selection for CHANGE PARAMETERS Submenu

X scale	specify maximum and minimum
Y scale	specify maximum and minimum
X ticks	specify the number of divisions in scale
Y ticks	specify the number of divisions in scale
Main title	type title for plot
X-axis title	type title for the X-axis
Y-axis title	type title for the Y-axis
X-axis decimals	specify the number of places shown to the right of the decimal for scale
Y-axis decimals	specify the number of places shown to the right of the decimal for scale

Table 10
Definitions of Items Found in the SHOW Menu

Color contours	Show/hide the color contours
Displacements	Show/hide the position of the nodes showing the deformed elements at the end of the current time step
Tensors	Show/hide a cross hair positioned at the center of each element that is scaled to show the orientation and magnitude of the principal stresses
Vectors	Show/hide an arrow drawn to scale from each node that shows the movement of the node during the time step

The size of the scaled quantities (displacements, tensors, and vectors) may be magnified or reduced by selecting the **Size** menu item. When the **Size** menu is selected and either **Size/Tensor**, **Size/Displacement**, or **Size/Vector** item has been selected a dialog box appears that has the parameter name in its title block. Within the box are a column of magnification/ reduction factors that are dimensioned in percent. The factors range from 10 to 1000. As the user selects a factor, the scaled quantity is redrawn in real-time response. The user may select any combination of factors that will provide an acceptable size for the scaled quantity. Once the user is satisfied, an "Ok" button located to the right of the factors should be selected to close the dialog box. The user may select the "Cancel" button to close the dialog box and return the size of the scaled quantities to the size in effect before the **Size** menu item was selected. Only one type of scaled quantity can be re-sized at any one time, although more than one of the scaled quantities can be visible having been turned on using **Show**.

Tensors are color-coded. White represents a positive (compressive) stress and black represents a negative (tensile stress). The tensors arrangement

also shows the orientation of the principal stresses. Vectors are shown as white arrows that are drawn from each node. The displacement, vectors, and tensors are drawn to a scale that maintains the proper relation in magnitude among like quantities regardless of the scale exaggeration selected using the **Size** menu.

The **Step** menu is a switch that loads the plot file data for one step each time the menu is selected. Selecting the menu with MB1 advances one step. Selecting the menu with MB2 backs up one step. Advancing past the final step returns to the initial step (T = 0.00).

30

5   Generic Use

Although XMESH was created expressly to work with STUBBS, the XMESH program has the ability to serve as a generic pre- and postprocessor. The information presented in Appendix A and Appendix B should provide enough guidance to enable a user to manipulate XMESH to create a mesh for a finite element program other than STUBBS. The user can also use XMESH to present the results of finite element analysis produced by programs other than STUBBS.

Preprocessing Considerations

All finite element programs known to the author require that the nodes be input with two dimensional coordinates. Usually the coordinates are Cartesian (x-axis and y-axis). In axisymmetric problems, the horizontal coordinate is based on a cylindrical coordinate system with the y-axis serving as the axis of symmetry. XMESH can provide a list of numbered nodes with two dimensional coordinates. It is in the element connectivity where finite element programs show diverse input requirements. STUBBS requires that the element nodes be entered in a counterclockwise direction. Also, STUBBS expects that the four corner nodes will be entered followed by four mid-side nodes. Where the mid-side nodes do not exist, zeros must be provided as place holders in the data. Programs other than STUBBS may use a clockwise convention for entering element connectivity. Also, some programs may require that the mid-side nodes be placed between the corner nodes in the listing of element nodes. The user may have to create an independent program to re-format the element connectivity data for the specific finite element program he plans to use.

Postprocessing Considerations

The user can modify the XMESH mesh file to create a plot file for postprocessing. An example of the modification procedure is presented in Appendix B Examples. The user must strip out the element and node parameters that are of interest from the output file of his finite element program. The parameters associated with nodes, that are calculated at nodal points, should be entered on the line of data containing the node number and x- and

y-coordinates for the node. The parameters calculated for elements should be entered on the line containing the element number. XMESH expects that each element has the horizontal normal stress, vertical normal stress, and horizontal shear stress on the vertical-right-hand face of the element entered on the line with the element number. Additional element parameters can be entered after the normal and shear stress on the same line.

The XMESH program will contour the parameters in color on the mesh. The element parameters can be assigned to the nodes by an interpolation scheme. The contours can be smoothed and displayed as a continuous spectrum of color on the mesh.

The XMESH program can show the displacement of the nodes and the displacement vectors of the nodes by a simple selection in the **Show** menu. The user can turn on a cross hair positioned at the center of each element that is scaled to show the orientation and magnitude of the principal stresses by using the **Show/Tensor** menu feature. The principal stresses and their orientation are automatically calculated by XMESH using the stress data found in the element data lines in the plot file. If the user has a program that does not calculate these stresses, it may be desired to zero out the stress data fields in the plot file. Additional element data can be displayed by XMESH. The data will be automatically contoured and may be assigned to nodes if this option is selected from the **Display/Data** menu.

Appendix A
File Structure of Mesh and Plot Files

The proper format for mesh files and plot files are shown in this appendix. More examples of mesh and plot files can be found in Appendix B Examples.

Structure of a Mesh File

The mesh file consists of *groupings* of line types.

Line of nodes and elements
A B C D E F G

where
A = number of nodes
B = number of solid elements
C = number of beam elements
D = number of slip elements
E = number of bar elements (reinforcing + anchor)
F = number of reinforcing bars
G = number of anchor bars

Example:

165 45 0 0 2 0 2

Comment: The mesh has 165 nodes, 45 solid elements, and 2 anchor bars.

Line of material types
H   I   J   K   L   M

where
H = number of solid material types
I = number of beam material types
J = number of slip material types
K = number of bar material types (rein. + anchor)
L = number of reinforcing bar material types

M = number of anchor bar types

Example:
2 0 0 1 0 1

Comment: The mesh has 2 solid material types and 1 anchor bar material.

List of material labels

Solid #1
.

.
Solid #H
Beam #1
.

.
Beam #I
Slip #1
.

.
Slip #n
Bar (Rein) #1
.

.
Bar (Rein) #L
Bar (Anchor) #1
.

.
Bar (Anchor) #M

List of nodes

nod e #	X-coord	Y-coord.	X-dof	Y-dof	pwp-dof
1	.1125	.47890	0	0	0
2	.2550	.4102	1	1	0
3	.5891	.0126	0	0	1
.					
.					
n					

Element connectivity
solid elements

el. #	mat#	4 cnr nodes				4 mid-side nodes				blocking code
1	1	1	4	3	2	0	0	0	0	0
2	1	4	6	10	11	0	0	0	0	1111
3	2	36	40	47	46	48	52	0	0	1001
4	2	12	11	8	22	21	30	34	35	0
.										
.										
.										
.										
m										

The mesh file can be displayed or refined within XMESH. A simple mesh file consisting of 4 eight-node elements is shown next.

```
21 4 0 0 0 0 0
 1 0 0 0 0 0
smat1
       1   .00000000E+00   .00000000E+00    0  0  0
       2   .25000000E+02   .00000000E+00    0  0  0
       3   .50000000E+02   .00000000E+00    0  0  0
       4   .50000000E+02   .25000000E+02    0  0  0
       5   .50000000E+02   .50000000E+02    0  0  0
       6   .25000000E+02   .50000000E+02    0  0  0
       7   .00000000E+00   .50000000E+02    0  0  0
       8   .00000000E+00   .25000000E+02    0  0  0
       9   .75000000E+02   .00000000E+00    0  0  0
      10   .10000000E+03   .00000000E+00    0  0  0
      11   .10000000E+03   .25000000E+02    0  0  0
      12   .10000000E+03   .50000000E+02    0  0  0
      13   .75000000E+02   .50000000E+02    0  0  0
      14   .10000000E+03   .75000000E+02    0  0  0
      15   .10000000E+03   .10000000E+03    0  0  0
      16   .75000000E+02   .10000000E+03    0  0  0
      17   .50000000E+02   .10000000E+03    0  0  0
      18   .50000000E+02   .75000000E+02    0  0  0
      19   .25000000E+02   .10000000E+03    0  0  0
      20   .00000000E+00   .10000000E+03    0  0  0
      21   .00000000E+00   .75000000E+02    0  0  0
       1   1    1    3    5    7    2    4    6    8    0
       2   1    3   10   12    5    9   11   13    4    0
       3   1    5   12   15   17   13   14   16   18    0
       4   1    7    5   17   20    6   18   19   21    0
```

Structure of a Plot File

The plot file structure consists of some lead lines that give the number of nodes, number of elements, and some information about data labels. The number of nodes and elements in the plot file must be consistent with those in the mesh file. If this data is not consistent then xmesh will inform the user that the mesh and plot files are incompatible. The plotting program expects x and y coordinates for each node. Additional node data can be provided. The program expects three stresses for each element, σ_x, σ_y, and τ_{xy}. Additional data can be provided for elements.

The number of additional data fields for nodes and elements is given in the first line of the plot file. The information provided is:

1st line

# of nodes	# of elements	Additional node fields	Additional element fields
xxx	xxx	nx	ex

example,

65	24	1	2

The next lines hold the labels that will be associated with the additional data fields.

next nx lines

    node field label (3) (fields 1 and 2 are expected to be x and y coordinates)

    node field label (4)

    .

    .

    node field label (nx)

next ex lines

    element field label (4) (fields 1, 2, and 3 are expected to be σ_x, σ_y, and τ_{xy}.)

    element field label (5)

    .

    .

    element label (ex)

Following the labels is a line that represents the time at the end of a time step. The node data follows the time declaration. The node data has one integer field and as many fields as will fit within 60 columns. The element data follows the nodes and has one integer field and as many real number fields as will fit within 60 columns.

A plot file that is compatible with the mesh file example is shown below.

```
 21 4 1 1
PWP
SF
T = 0.00
        1   .00000000E+00   .00000000E+00      0
        2   .25000000E+02   .00000000E+00      0
        3   .50000000E+02   .00000000E+00      0
        4   .50000000E+02   .25000000E+02      0
        5   .50000000E+02   .50000000E+02      0
        6   .25000000E+02   .50000000E+02      0
        7   .00000000E+00   .50000000E+02      0
```

8	.00000000E+00	.25000000E+02	0
9	.75000000E+02	.00000000E+00	0
10	.10000000E+03	.00000000E+00	0
11	.10000000E+03	.25000000E+02	0
12	.10000000E+03	.50000000E+02	0
13	.75000000E+02	.50000000E+02	0
14	.10000000E+03	.75000000E+02	0
15	.10000000E+03	.10000000E+03	0
16	.75000000E+02	.10000000E+03	0
17	.50000000E+02	.10000000E+03	0
18	.50000000E+02	.75000000E+02	0
19	.25000000E+02	.10000000E+03	0
20	.00000000E+00	.10000000E+03	0
21	.00000000E+00	.75000000E+02	0
1	0.33	1.05	-.045	1.15
2	0.33	1.05	.045	1.15
3	0.16	.56	-.67	2.28
4	0.16	.56	.67	2.28

T = 0.02

1	.00000000E+00	.00000000E+00	0
2	.25000000E+02	.00000000E+00	0
3	.50000000E+02	.00000000E+00	0
4	.50000000E+02	.24000000E+02	0
5	.50000000E+02	.47000000E+02	0
6	.25000000E+02	.47000000E+02	0
7	.00000000E+00	.47000000E+02	0
8	.00000000E+00	.24000000E+02	0
9	.75000000E+02	.00000000E+00	0
10	.10000000E+03	.00000000E+00	0
11	.10000000E+03	.24000000E+02	0
12	.10000000E+03	.47000000E+02	0
13	.75000000E+02	.47000000E+02	0
14	.10000000E+03	.71100000E+02	0
15	.10000000E+03	.08000000E+03	0
16	.75000000E+02	.08000000E+03	0
17	.50000000E+02	.08000000E+03	0
18	.50000000E+02	.71100000E+02	0
19	.25000000E+02	.08000000E+03	0
20	.00000000E+00	.08000000E+03	0
21	.00000000E+00	.71100000E+02	0
1	0.53	1.95	-.05	1.05
2	0.53	1.95	.05	1.05
3	0.46	.96	-1.07	2.2
4	0.46	.96	1.07	2.2

Appendix B
Example: Finite Element Development

This Appendix contains an example of a MESH file during the creation and refinement of the mesh. The series of files named fun*.msh illustrate the changes in the mesh file produced by routine tasks. File fun1.msh was created by starting XMESH and using **File/New**. Table B1 shows the steps used in creating the mesh file beginning with the prompt produced by the selection of **File/New**.

Table B1		
Steps for Mesh Development		
Command Line	**Message Line**	**User Actions**
New filename	Enter a filename for the new mesh	fun1.msh
Mesh coordinates, lower left, X, Y:_	Enter the extreme coordinates of the mesh.	-10,-10
Mesh coordinates, upper right - X,Y:_	Enter the extreme coordinates of the mesh.	110,110
Enter material type number:_	Material Properties for Solid Elements	1
Enter material type title:_	Material Properties for Solid Elements	smat1
Select approx. center of element- MB1 (MB3 to quit):_	Material Properties for Solid Elements	50,50 (typed)(could have used mouse)
Select corner of the new element - MB1 (MB3 to quit):_	Material Properties for Solid Elements	used MB1 to click on lower left corner
Select location for this corner - MB1 (MB3 to quit):_	Material Properties for Solid Elements	0,0 (typed) (could have used mouse)
		(Continued)

Table B1 (Concluded)

Command Line	Message Line	User Actions
Select corner of the new element - MB1 (MB3 to quit):_	1: Node at (.0,.0)	used MB1 to click on lower right corner
Select location for this corner - MB1 (MB3 to quit):_	1: Node at (.0,.0)	100,0
Select corner of the new element - MB1 (MB3 to quit):_	2: Node at (100.0, .0)	used MB1 to click on upper right corner
Select location for this corner - MB1 (MB3 to quit):_	2: Node at (100.0, .0)	100,100
Select corner of the new element - MB1 (MB3 to quit):_	3: Node at (100.0, 100.0)	used MB1 to click on upper left corner
Select location for this corner - MB1 (MB3 to quit):_	3: Node at (100.0, 100.0)	0,100
Select approx. center of element- MB1 (MB3 to quit):_	4: Node at (.0, 100.0)	used MB3 to quit

Immediately after MB3 was pressed, the element was redrawn to nearly fill the screen. The command line and message area were erased. A screen dump of the mesh is shown as Figure B1.

Select **File/Save**.
Command line prompts, "Mesh filename:_"

The filename was entered as fun1.msh (CR). The file was saved.
The contents of the file are shown below.

```
4 1 0 0 0 0 0
1 0 0 0 0 0
smat1
     1    .00000000E+00    .00000000E+00    0  0  0
     2    .10000000E+03    .00000000E+00    0  0  0
     3    .10000000E+03    .10000000E+03    0  0  0
     4    .00000000E+00    .10000000E+03    0  0  0
     1    1       1       2     3      4     0    0    0    0   0
```

The file is organized as described in **Appendix A**. The first line has the number of nodes and elements. The five zero's that follow the number of nodes (4) and solid elements (1), represent that numbers of beam elements, slip elements, bar elements (total), anchor bars, and reinforcing bar elements. The second line indicates that there is one solid element material label to follow and no beam, slip, and bar material labels. The third line is

the name (or label) of the solid material. The next 4 lines are node numbers, coordinates (X,Y) and the codes for degrees of freedom (dof) for displacements (X,Y) and pore water pressure (pwp). The next line shows the element number, solid material number, the four corner nodes, the four mid-side nodes an the blocking code. The nodes are written in a counterclockwise direction from the first node listed. The mid-side nodes are likewise entered in a counter-clockwise direction starting with the mid-side node on the side whose ends are the first two listed corner nodes. The blocking code is an integer that specifies which sides may divided during an **Edit/Refine** operation. It is easy to change the blocking code within XMESH. If a hand-made file is to be loaded into XMESH it is recommended that a zero blocking code be used in preparing the file.

At this time with the file still loaded in XMESH, the menu item **Edit/Refine** was selected. The program refined the mesh by quartering the initial element, yielding a total of 9 nodes and 4 elements. The refined mesh was saved as fun2.msh and the data file is shown below. A screen dump of the mesh is shown as Figure B2.

```
9 4 0 0 0 0 0
1 0 0 0 0 0
smat1
    1    .00000000E+00    .00000000E+00       0  0  0
    2    .50000000E+02    .00000000E+00       0  0  0
    3    .50000000E+02    .50000000E+02       0  0  0
    4    .00000000E+00    .50000000E+02       0  0  0
    5    .10000000E+03    .00000000E+00       0  0  0
    6    .10000000E+03    .50000000E+02       0  0  0
    7    .10000000E+03    .10000000E+03       0  0  0
    8    .50000000E+02    .10000000E+03       0  0  0
    9    .00000000E+00    .10000000E+03       0  0  0
    1    1    1    2    3    4    0    0    0    0    0
    2    1    2    5    6    3    0    0    0    0    0
    3    1    3    6    7    8    0    0    0    0    0
    4    1    4    3    8    9    0    0    0    0    0
```

The mesh file can be easily modified to become a plot file that may be used in postprocessing with XMESH. The conversion can be accomplished in two steps. First, the data not needed in postprocessing are stripped from the mesh file. Second, the results of the analysis added to the node and element data. Then, the mesh file can be loaded into XMESH and the plot file can be loaded using the **File/Load** data menu item.

Shown below is the mesh file FUN2.msh with the data unnecessary to the postprocessing stripped away.

```
9 4
    1    .00000000E+00    .00000000E+00
    2    .50000000E+02    .00000000E+00
    3    .50000000E+02    .50000000E+02
```

```
4    .00000000E+00    .50000000E+02
5    .10000000E+03    .00000000E+00
6    .10000000E+03    .50000000E+02
7    .10000000E+03    .10000000E+03
8    .50000000E+02    .10000000E+03
9    .00000000E+00    .10000000E+03
1
2
3
4
```

Then, data from the analysis is added to the stripped-down mesh file. The plot data expected by XMESH is composed of the x- and y- coordinates of the node after each time step. Three quantities are expected for the element data. They are σ_x, σ_y, and τ_{xy}. Additional node and element parameters may be plotted. On the first line of the plot file, after the number of nodes (9) and number of elements (4), XMESH expects two integers. The first integer is the number of additional node parameters to be plotted and the second integer is the number of additional element parameters to be plotted. If no additional parameters were to be plotted, the first line in the plot file should be:

9 4 0 0

If two additional node parameters and one additional element parameter were to be plotted, the first line of the plot file should be:

9 4 2 1

If any additional parameters are to be plotted, the labels of these parameters should be provided one on a line per parameter.

Lets us say that pore water pressure and temperature were the additional node parameters and safety factor was the additional element parameter. The first lines of the plot file should be

9 4 2 1
PWP
T
SF

The labels for the parameters are abbreviated, because limited space is provided for the parameter names on the dialog box where the parameters are selected for display.

Following the labels (if any) are the node and element data. The node and element data must be preceded by a line that provides the time associated with the end of the time step. Usually, the initial time is zero.

The time data line should appear as:

T = 0.0

Then, the node and element data follow. Data for additional times should be added at the end of the element data. Finally, the data are terminated by a END line.

The plot file with three time steps and one additional node parameter is shown below:

```
 9  4  1  0
pwp
T = 0.00
      1   .00000000E+00   .00000000E+00   0.00
      2   .50000000E+02   .00000000E+00   0.00
      3   .50000000E+02   .50000000E+02   0.00
      4   .00000000E+00   .50000000E+02   0.00
      5   .10000000E+03   .00000000E+00   0.00
      6   .10000000E+03   .50000000E+02   0.00
      7   .10000000E+03   .10000000E+03   0.00
      8   .50000000E+02   .10000000E+03   0.00
      9   .00000000E+00   .10000000E+03   0.00
   1     12          30        -6
   2     12          30         6
   3     20          45       -15
   4     20          45        15
T= 0.10
      1   .00000000E+00   .00000000E+00   1.0
      2   .50000000E+02   .00000000E+00   1.5
      3   .50000000E+02   .40000000E+02   1.2
      4   .00000000E+00   .40000000E+02   1.5
      5   .10000000E+03   .00000000E+00   1.1
      6   .10000000E+03   .40000000E+02   1.0
      7   .10000000E+03   .88000000E+02   1.4
      8   .50000000E+02   .88000000E+02   1.5
      9   .00000000E+00   .88000000E+02   1.3
   1     12    30    -6
   2     12    30     6
   3     22    50   -15
   4     24    55    16
T = 1.0
      1   .00000000E+00   .00000000E+00    1.2
      2   .50000000E+02   .00000000E+00    1.6
      3   .50000000E+02   .30000000E+02    1.0
      4   .00000000E+00   .30000000E+02    0.3
      5   .10000000E+03   .00000000E+00    0.3
      6   .10000000E+03   .25000000E+02   -0.4
      7   .10000000E+03   .79000000E+02   -0.4
      8   .50000000E+02   .77000000E+02    0.0
      9   .00000000E+00   .83000000E+02    0.0
```

```
1    10    30   -7
2    10    30    7
3    30    60  -12
4    33    65   14
END
```

No comment lines are permitted in these files.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE April 1997	3. REPORT TYPE AND DATES COVERED Final report

4. TITLE AND SUBTITLE
Guide to XMESH

5. FUNDING NUMBERS

6. AUTHOR(S)
John F. Peters, Ronald E. Wahl, Ronald B. Meade, Raju Kala

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
U.S. Army Engineer Waterways Experiment Station
3909 Halls Ferry Road, Vicksburg, MS 39180-6199
Mevatec Corporation
3046 Indiana Avenue, Suite 172
Vicksburg, MS 39180

8. PERFORMING ORGANIZATION REPORT NUMBER
Miscellaneous Paper GL-97-6

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U.S. Army Corps of Engineers
Washington, DC 20314-1000

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
Available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA 22161.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The Computer program XMESH, is a finite element preprocessor, postprocessor, and a menu-driven input file builder. The program was developed at the U.S. Army Engineer Waterways Experiment Station as a productivity tool to be used with the finite element program, STUBBS. XMESH runs on any UNIX based system which has the software packages XWINDOWS and MOTIF. Although XMESH was written specifically for use with STUBBS, the program can be used with any finite element program as a preprocessor to develop meshes through the creation of nodes and elements and as a postprocessor to provide a visual display of the results of the analysis.

14. SUBJECT TERMS
Finite element method
Postprocessor
Preprocessor

15. NUMBER OF PAGES
52

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED	18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT